

## **Computation of High Speed Cold Cavity Flow with a Commercial CFD Code**

Ashfaque A. Khan and T. R. Shembharkar

Propulsion Division, National Aerospace Laboratories, Bangalore.

### **ABSTRACT**

A commercial CFD code based on a pressure-based algorithm (SIMPLEC) has been used to compute high Mach number flow over a cavity, which has application in supersonic combustors. The difficulties encountered in obtaining converged solution for compressible flow by the pressure-based method are discussed. The predicted solution is compared with the available experimental results. Though some quantitative differences exist between them, the qualitative agreement is generally good.

Keywords: CFD, cavity, supersonic flow

### **1. INTRODUCTION**

Supersonic air-breathing engines are vital elements of future high-speed transportation vehicles. These engines invariably include supersonic combustors. In a scramjet, flame holding with minimum total pressure loss is a key to obtain good performance. In recent years, cavities have been proposed as a new concept for flame holding and stabilization in supersonic combustors. Therefore, it is crucial to understand the basic flow features of cavities in high-speed flows. There have been some excellent review papers [Ben-Yakar et al] on the aspects of high-speed flow behaviour over the cavity. In general, the boundary layer ahead of the cavity separates at the leading edge and forms a free shear layer across the cavity. Inside the cavity, flow recirculation takes place. The shear layer reattaches at some other point downstream. The reattachment point depends upon the geometry of the cavity and the external flow conditions. Depending upon the reattachment point, the cavities are classified as "open" or "closed". In open cavity, the reattachment takes place at the back face of the cavity and in closed cavity it takes place on the lower wall. The open cavities have aspect ratio less than about 7-10 while the closed cavity have higher aspect ratios. The leading edge expansion wave, flow separation, flow recirculation, reattachment of free

shear layer and trailing edge shock contribute to the pressure loss in the cavity which needs to be kept low for acceptable performance.

The present investigation is related to flow analysis through a slant cavity (the back face is inclined), which can be used in a scramjet to get the flame holding and recirculation properties. The analysis has been carried out using a commercial CFD code called CFD-ACE+ [CFDRC], which is a general purpose CFD code based on a pressure-based algorithm. Besides understanding the cavity flow, there is a second motivation for carrying out this work. In CFD literature, there appears to be a sort of classification of numerical methods namely, pressure-based or density-based, depending upon how the continuity equation is treated. In purely incompressible flow domain, density being constant and not a function of pressure, there is no separate equation for pressure. The Poisson-type pressure or pressure correction equation is derived from continuity and momentum equations. Such methods are broadly called pressure-based methods. In compressible flow domain, the continuity equation becomes the transport equation for density and is solved separately for density. Pressure is then obtained from some relation between density and pressure e.g. ideal gas law. Such methods are broadly called density-based methods. Both these methods have their plus and minus points. The pressure-based methods are very robust in incompressible and low Mach number flows while the density-based methods are well suited to compressible flows. The pressure-based methods seem to get into trouble in highly compressible flow regime while the density-based methods face severe problems of convergence in incompressible flow regime. It has been a very active area of research to extend the working flow regimes of these two approaches by devising innovative strategies. In the present CFD scenario, when commercial CFD codes are increasingly being used for solving practical industrial problems, it may be noted that most of these codes are based on one or the other method mentioned above. The code CFD-ACE+, which we have used, is based on a pressure-based method. Our goal was to explore the applicability or otherwise of the pressure-based algorithm for high speed flow situation under consideration here. The computation has been performed for a given slant cavity configuration at the inlet Mach number of 2.46 and the predicted results have been compared with the experimental results [Samimy et al].

## 2. COMPUTATIONAL DETAILS

Figure-1 shows the computational domain and the geometry of the slant cavity. A 2-D multi-block grid with 16233 cells was generated. The grid distribution was prescribed keeping in mind the expected flow features and flow gradients. The 2-D steady state RANS equations were solved along with RNG k- $\epsilon$  turbulence model. The present computations have been done with a pressure-based method (SIMPLEC algorithm) on a body-fitted structured grid. It is a cell centered finite volume approach. All the dependent and auxiliary variables are stored at the cell center. The convective terms are discretized by upwind scheme and the diffusion terms by central difference scheme. An iterative segregated solution method is employed, wherein the equation

sets for each variable are solved sequentially and repeatedly by conjugate gradient squared with pre-conditioning linear solver until a converged solution is obtained. The dependent variables are under-relaxed by inertial under-relaxation (false time-step) and the auxiliary variables by linear under-relaxation to ensure overall convergence.

The relevant velocity, pressure and density at the air inlet plane were taken from Reference [Samimy et al]. The inlet profiles were taken as uniform. All the bottom walls of the cavity were specified no-slip wall condition. The upper boundary was, however, given symmetry condition, which effectively means inviscid sliding impervious wall. The exit boundary conditions were satisfied by extrapolating the corresponding variables from inside.

### 3. RESULTS AND DISCUSSION

During the computations, it was realized that the high-speed flow computation with the code was fraught with difficulties. Unlike the incompressible and low Mach number compressible flow regimes for which the code has shown extremely robust character [Muralidhara et al], the high Mach number flow regime makes it vulnerable to oscillations and divergence. In the present computation, the inlet conditions if prescribed in terms of total pressure and total temperature at the inlet, the solution could not be obtained. However, the equivalent inlet conditions in terms of velocity and density at inlet stabilizes the convergence. Further, the no-slip wall condition on the top boundary also made the solution prone to divergence. The higher order differencing for convective terms also posed problems in obtaining converged solution. In short, it required extensive experimentation with different options to get some meaningful solution for the problem. This experience supports the widely held opinion about the pressure-based algorithms that they are susceptible to non-convergence and divergence of solution in high Mach number flow regime. The final solution was obtained with prescribed velocity and density conditions at the inlet, symmetry condition at the top boundary, first order upwind scheme for convective terms and the RNG k- $\epsilon$  turbulence model. This is the solution which is presented in the paper and compared with the experimental results [Samimy et al].

The predicted solution, in spite of slight difference in top wall boundary condition, is in qualitative agreement with the experimental observations. The leading edge expansion wave, low Mach number flow region inside the cavity, free shear layer above the cavity, reattachment of shear layer on the slant wall and an oblique shock downstream (recompression and shock) are clearly visible in Figure-2. The details of the recirculation zone can be seen in the Figure-3. The position of the center of recirculation compares favourably with the experimental observation [Samimy et al]. A comparison of predicted and measured location of  $M=1$  line (sonic line) and  $U=0$  line (dividing line, some measure of recirculating vortex) in the flow domain is shown in Figure-4. The predicted sonic line is slightly below the experimental value over the cavity but matches well along the slant wall. Similarly the dividing line  $U=0$  is below the experimental line but the difference is considerably reduced near slant wall.

Figure-5 shows the non-dimensional velocity profiles at different streamwise locations in the flow field. These profiles also show good qualitative agreement with the experimental results shown here. Figure-6 compares the wall static pressure in the cavity. The present predictions are the static pressure at the bottom of the cavity. They seem to be slightly lower than the experimental values inside the cavity and slightly higher along the slant wall. It may be noted here that the exact location of the measurement points for static pressure is not quite clear in the reference.

#### 4. CONCLUSIONS

The commercial CFD code, namely CFD-ACE+, based on a pressure-based algorithm has been used to compute flow over a cavity with free stream Mach number of 2.46. It is concluded that the pressure-based algorithms do face problems of convergence in high Mach number flows. It is not very straightforward to use this algorithm for compressible flows. It needs quite some adjustment with various solution controlling options and boundary conditions to get meaningful solutions. The predicted solution is compared with the available experimental results. Though some quantitative differences exist between them, the qualitative agreement is generally good. The expansion wave at the leading edge, flow separation, flow recirculation inside the cavity, reattachment of free shear layer at the slant wall and the trailing edge shock have been qualitatively well predicted.

#### 5. ACKNOWLEDGEMENT

The authors thank the CSIR Centre for Mathematical modelling and Computer Simulation (C-MMACS), Bangalore for providing the computing facilities.

#### REFERENCES

Adela Ben-Yakar and Ronald K. Hanson, 'Cavity flame-holders for ignition and flame stabilization in scramjets: An overview', *Journal of Propulsion and Power*, 17(4), 869-877, 2001.

CFD-ACE+ code from CFD Research Corporation, USA ([www.cfdrc.com](http://www.cfdrc.com)) – available to the authors courtesy Centre for Mathematical Modelling and Computer Simulation (C-MMACS), Bangalore.

Samimy, M., Patrie, H. L. and Addy, A. L., "A study of compressible turbulent reattaching free shear layers", *AIAA Journal*, 24(2), 261 (1986)

H.S.Muralidhara, B.B.Chinmaya Kumar and T.R.Shembharkar, "CFD analysis of gas turbine combustor: A CAD to Mesh to CFD approach", 6<sup>th</sup> National Conference on Air Breathing Engines and Aerospace Propulsion (NCABE), Paper B1.1, 10-11 January 2003.

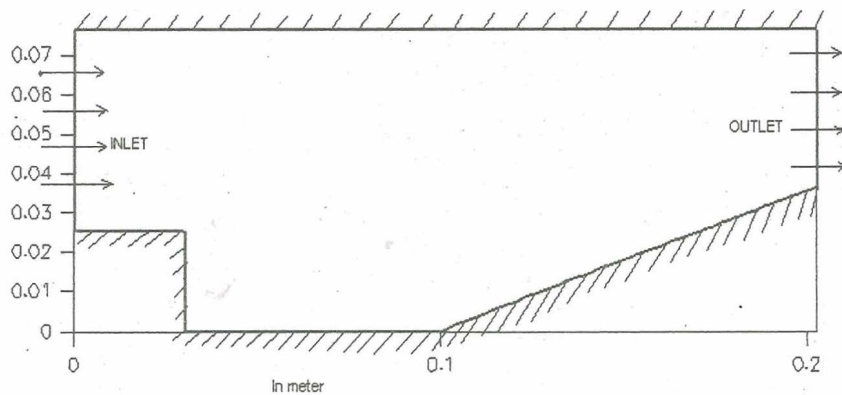


Figure 1. Computational Domain for the Slant Cavity.

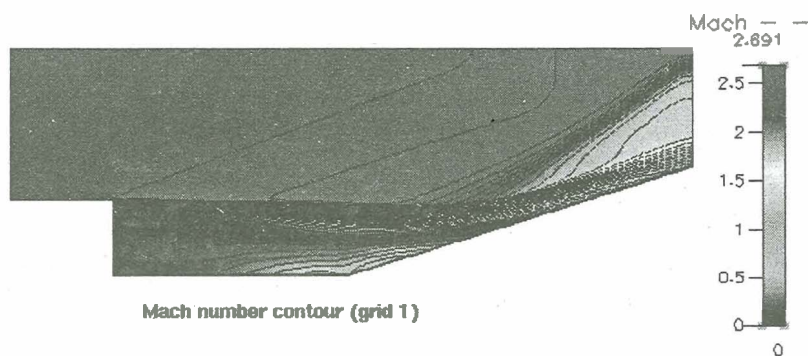
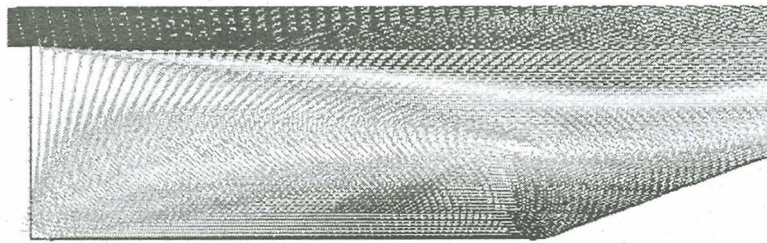


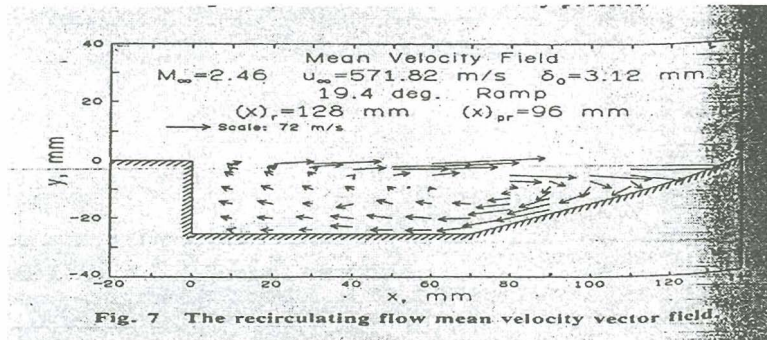
Figure 2. Mach Number Contours (Present Predictions).





U velocity vector field (grid 1)

Present Predictions.



Samimy et al.

Figure 3. Recirculating Flow in the Cavity (Present Predictions and Samimy et al).

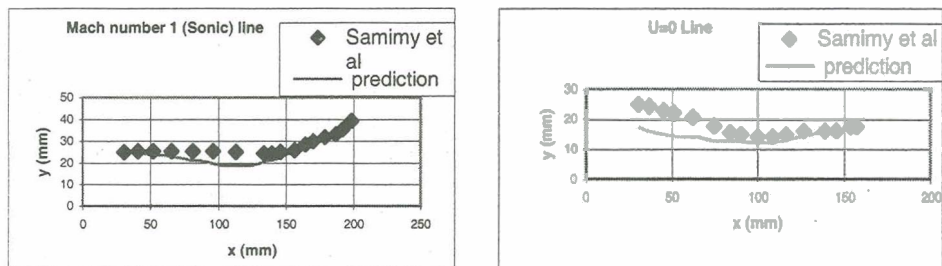
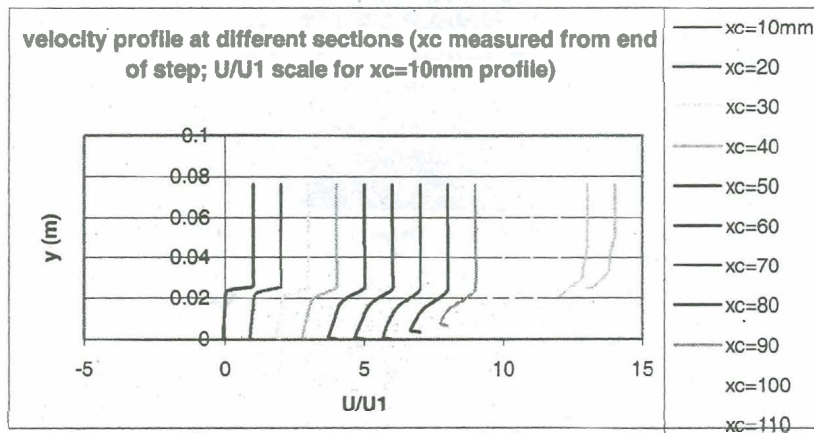
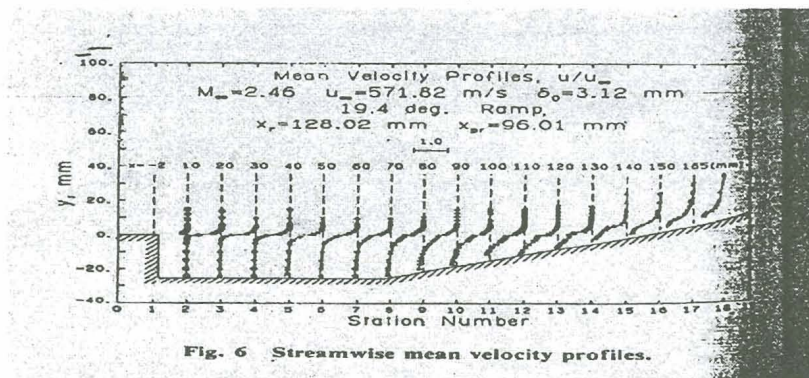


Figure 4. Comparison between Present Predictions and Experimental Results for Sonic and U=0 line.



Present Predictions.



Samimy et al.

Figure 5. Velocity Profiles at different Sections (Present Predictions and Samimy et al).

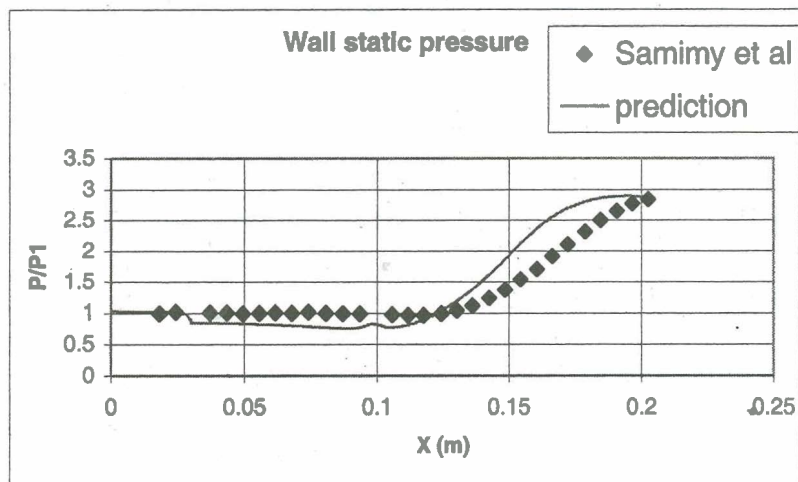


Figure 6. Comparison between Present Predictions and Experimental Results for Wall Static Pressure.